8411

p -37

ADVANCING COMPUTATIONAL AERODYNAMICS THROUGH

WIND-TUNNEL EXPERIMENTATION

JOSEPH G. MARVIN
Ames Research Center, NASA
Moffett Field, California 94035, U.S.A.

AGARD FLUID DYNAMICS PANEL
Subcommittee on Wind Tunnels & Testing Techniques

MEETING ON INTEGRATION OF COMPUTERS AND WIND TUNNEL TESTING
September 24-25, 1980

ADVANCING COMPUTATIONAL AERODYNAMICS THROUGH

WIND-TUNNEL EXPERIMENTATION

Joseph G. Marvin

Chief, Experimental Fluid Dynamics Branch

Ames Research Center, NASA

Moffett Field, California 94035, U.S.A.

SUMMARY

The rapid development of Computational Fluid Dynamics from flow-field solutions governed by simple inviscid flow equations to those governed by complex viscous Navier-Stokes flow equations is briefly reviewed. Engineering applications in this rapidly developing environment must be accompanied by a careful assessment through comparison with experiment. A framework for complementary experimentation that can verify and critically assess the development of CFD is outlined. Experiments are categorized broadly as phenomenological and configurational, and it is proposed that they be keyed directly to the developmental stages of CFD codes. A building-block concept that couples experiment and computation is introduced. It is based on the idea of providing carefully documented modeling information for research-code development and carefully documented verification data for pilot codes that efficiently extend the research codes beyond their original demonstrative conditions. Building-block experiments, designed to supply turbulence modeling information for certain flow phenomena encountered in aerodynamic applications, are outlined. Some results

from these experiments and their complementary computations are used to illustrate the synergism of this concept.

SYMBOLS

b	span length
c	chord length
c_f	skin friction coefficient
C_D	drag coefficient
C_F	average skin-friction coefficient
C_h	Stanton number
C_H	average Stanton number
C_L	lift coefficient
C_M	moment coefficient
C_p	pressure coefficient
C_p^*	pressure coefficient when local Mach number is sonic
H	tunnel height
L	body length
M	Mach number
p_w	wall pressure
P_t	total pressure
q	heating rate
q_o	stagnation heating rate
Re	Reynolds number
t	thickness
$\bar{t}$	time
T_t	total temperature

T_w	wall temperature
u	velocity in the free-stream direction
W	tunnel width
x	distance along body in free-stream direction
x_o	distance to first rise in pressure in vicinity of interaction zone
y	distance along body in a direction normal to a centerline along the x-axis
α	angle of attack
δ_o	boundary-layer thickness at x_o
η	fraction of semispan
Δp	incremental change from local pressure
Λ	sweep angle
ρ	density

Subscripts:

i,j	direction indices
∞	free stream
l	local value
s	shock wave

Superscript:

$(\bar{\ })$	mean valve

INTRODUCTION

Computational Fluid Dynamics (CFD) is recognized as having a significant role in the future development of aerospace vehicles (Refs. 1, 2). It provides important, new technological capability. For example, flow-field simulations in wind tunnels are often limited by model size, air speed, density, temperature, and undesirable effects from walls and stings. Such limitations often contribute to uncertainties in extrapolation of data to flight Mach numbers and Reynolds numbers. CFD is limited ideally only by requirements of computer speed and storage and therefore is an attractive means of providing that necessary bridge between wind-tunnel simulation and flight. Also it provides economic incentives that can reduce the cost and time requirements for developing new vehicles by providing important preliminary information on optimum configurations, now achieved by cut-and-try wind-tunnel testing, and by providing simulations for incremental changes in design values of speed, altitude, and angle of attack, also achieved by time consuming wind-tunnel testing.

Much work remains before the full advantages of CFD can be realized, and future requirements of computer speed and storage must be met (Ref. 2). Equally important, however, is the careful development and critical assessment of CFD through comparison with experiment. It is only through the latter that confidence can be achieved regarding the ability to simulate flows a priori. The wind tunnel provides an excellent means for accomplishing this. In fact, future aerospace vehicle development will require very close coupling between CFD and wind-tunnel simulations.

4

An examination of data sets from wind-tunnel experiments reveals that only a few are suitable for evaluating the new emerging CFD codes. See, for example, Ref. 3, in which flows in the transonic flight regime were examined. The root cause of the deficiencies in the experiments arises mainly because their original purpose was resolution of an engineering problem or an attempt to improve performance of a configuration or concept and not because they were rife with incorrect data. Only recently have data sets begun to emerge that were generated for the purpose of improving computational fluid dynamics (Ref. 4). At this particular time what is needed are more carefully planned experiments that can assess the predictive capabilities for a variety of aerodynamic flows.

This paper proposes a framework for complementary experimentation that can expedite, verify, and critically assess the development of CFD. Suggestions are made for categorizing experiments in a logical sequence that paces itself with the development of CFD. Discussion regarding test condition ranges and the importance of boundary conditions is presented. Flow measurement requirements are introduced according to the degree of sophistication required for code assessment. The paper concludes with a discussion of the status and plans of an experimental program now under way.

STATUS OF CFD

In the last decade, computational power has increased significantly and real advances in CFD have been possible (Ref. 1). The status is summarized in Table 1 where the stages of governing equations and their

corresponding approximations are listed. In the past, flow simulations were provided by invoking Prandtl's ideas and solving the linearized inviscid equations and estimating viscous effects subsequently through analytic or momentum integral solutions of the boundary layer. At high Reynolds numbers, the Reynolds-averaged form of the equations was applied and turbulence modeled accordingly. In the early 1970s, inviscid solutions to the nonlinear inviscid equations began to emerge. Presently, these equations are being solved routinely with viscous effects accounted for by coupling the viscous boundary-layer equations to the inviscid solution. Examples are given in Figs. 1 and 2a that illustrate present capabilities and the use of wind-tunnel data in assessing those capabilities.

The surface heating distribution on the windward surface of a model of the Space Shuttle vehicle is shown in Fig. 1a. Predictions (Ref. 5) of the windward surface heating obtained by coupling a three-dimensional inviscid solution with an approximate three-dimensional boundary-layer method are shown for the most windward streamline and two spanwise strips. The data (Ref. 6) are shown to substantiate the predictions and provide a measurable degree of confidence in the extrapolation to full scale and flight speeds. The latter aspect is essential to the development of this vehicle because no ground-based facilities are available that can simulate the real combination of its speed and scale, both of which are important factors in determining vehicle heating. This is in contrast to blunt capsule-type entry vehicles for which the well-known Mach number freeze is applicable and for which wind-tunnel model heating distributions can be extrapolated to flight directly by scaling with

the stagnation point reference heating. Figure 1b shows the prediction of the flight heating distribution for a point along the vehicle trajectory using the appropriate real gas chemistry in the computer code that was verified by wind-tunnel data. A comparison with the wind-tunnel distribution is given so as to emphasize that there are significant differences between the two in the forebody region. These differences mean that wall temperatures can be 200°R higher. The differences arise because the local boundary-layer edge velocities are much higher at the flight conditions due to speed, scale, and gas composition differences and to corresponding differences in entropy layer swallowing.

An example of transonic flow prediction intended to result in a shockless supercritical airfoil design (Ref. 7) is given in Fig. 2a. The conditions are for an off-design case sensitive to large viscous effects. The solid line is the prediction using the nonconservative inviscid equations used in Ref. 8, and the dashed line is a subsequent solution corrected for viscous, turbulent boundary-layer displacement effects without any consideration of local separation and obtained using the integral methods of Ref. 9. The data are shown to substantiate the method employing inviscid-viscid coupling.

The foregoing examples indicate the present uses of CFD in design applications. It is not the author's intent to suggest that the state of affairs in the use of interactive techniques always results in successful comparisons with data. In fact, much work is under way to perfect these techniques (Refs. 10, 11). Obviously, their biggest shortcoming is in the case of flow separation.

The next step in the development of CFD will be the application of the full Navier-Stokes equations in their Reynolds-averaged form and with modeled turbulence. In this approach, inviscid-viscid interactions, including separation, are captured simultaneously. Already, good progress has been made toward developing such methods (Refs. 12, 13). Of course, their computation times are still excessive and their routine use in practical applications awaits a larger, faster computer and more efficient algorithms (Ref. 2).

Figure 2b shows a comparison of a solution from a Navier-Stokes code (Ref. 14) with the interactive code of Ref. 9 and with the experimental data for the supercritical airfoil discussed previously. Agreement with the viscous interactive code and the data is quite good.

The advantage of the Navier-Stokes code is realized when angle of attack is increased and maximum lift and associated shock-boundary-layer separation occur. An example is illustrated in Fig. 3 in which the drag polar and lift versus angle-of-attack curves from various computations are compared with the experiment. For this particular airfoil a buffet domain beyond an angle of attack of 3° is computed. Examination of the data also indicated buffet, but at slightly higher angles of attack. Later, this important predictive capability will be discussed further. Data are given for two wind-tunnel wall porosities and the indication is that the data are not entirely free of boundary effects. Nevertheless, the predicted trends from the Navier-Stokes solutions are in reasonable agreement with the data and are much better than the trends predicted by nonlinear inviscid predictions.

In the far future, simulations using the full Navier-Stokes equations without Reynolds averaging are anticipated; research into this aspect of CFD is ongoing (Ref. 15). Practical engineering use of these simulations may not be possible without much larger computers, but their application toward understanding fluid physics and turbulence modeling for Reynolds-averaged codes is recognized (Ref. 16). Experimentation is also needed to verify and advance this future effort in CFD, but that will not be addressed in this paper.

EXPERIMENTAL REQUIREMENTS

The examples described above illustrated some of the present and future capabilities of CFD. In each case, experimental data were used to substantiate the computations. However, uncertainties in boundary conditions (e.g., Fig. 3) and lack of other detail makes it difficult to completely assess the validity of computations (Ref. 3). In this section a framework for complementary experimentation that can expedite, verify, and critically assess the development of CFD is outlined.

Test Conditions

The Mach number and Reynolds number domain for aerospace vehicles is shown in Fig. 4. Mach number varies from subsonic to hypersonic, encompassing the range encountered by commercial passenger vehicles, high-performance military aircraft, and NASA's Space Shuttle. Clearly, the range of interest encompasses Reynolds numbers at which the flow will be turbulent and also at which our facility capabilities are limited with respect to complete configuration testing. It is necessary,

however, that experiments be performed over this range to critically examine the adequacy of CFD and to establish confidence in its extrapolative capability. This apparent dilemma can be circumvented by compromise in the actual test conditions for complete configurations and by careful planning of other experiments that test the ability of the codes to predict critical phenomena over the complete range.

Categorization of Experiments

The division of experiments into phenomenological and configuration categories can be keyed directly to the development of CFD codes as illustrated in Fig. 5. The codes and corresponding experiments have been divided into three separate developmental stages, although it is recognized that very often overlap between stages exists. The first stage of CFD development is the research phase where, for example, the ability to predict certain flow phenomena might be established. Experiments of the building-block variety, which provide phenomenological modeling information, are needed at this stage. Next, more efficient pilot codes are developed to extend the research codes' applicability to a wider range of conditions or to different geometries. Verification experiments, which provide parametric information, are needed at this point. Beyond this stage, production codes are provided for routine design applications. Design experiments, which address optimal configurational performance, are needed at this stage. Properly planned, this synergistic approach should accelerate the development process.

10

Elements of Well-Documented Experiments

Each of the experimental stages must provide specific information for guidance and critical assessment of CFD. Key elements for each category of experiment are listed in Table 2. Building-block experiments must document sufficient information on flow phenomena to provide guidance in flow modeling and to provide a critical test of the codes' performance once the modeling is established. Surface variables and flow variables, including turbulence information, are essential, and measurements are required at test conditions representative of flight Mach numbers and Reynolds numbers. Verification experiments must provide sufficient information to test the ability of pilot codes to perform adequately over a range of flow conditions or for a variety of configurations. At this stage in the development, detailed information on flow modeling is not essential, but parametric testing over the full range of flight Mach and Reynolds numbers is essential. Design experiments provide the optimal configuration data necessary for design performance evaluation and should be carried out as close to flight conditions as practical.

The Building-Block Concept

A building-block concept for developing flow models is outlined in Fig. 6. The idea is to divide complex flows into a series of isolated problems that deal individually with certain flow phenomena. Building-block experiments and companion computations are then carried out to provide guidance and critical assessment of the research codes. The concept also applies to pilot code development, but the experiments must

11

be performed over appropriate parametric ranges or configuration changes. It should be emphasized that this particular concept does not preclude experimental or computational discoveries of new physical phenomena. In fact, this synergistic approach can accelerate discovery and in-depth study of new phenomena, otherwise passed off as fortuitous or unexplained measurements or as spurious or nonconverged computer solutions. An example will be discussed subsequently.

Test Boundary Conditions

As noted in Table 2, boundary conditions must be documented for experiments keyed to the development of CFD. The importance of specifying far-field boundary conditions arises because these conditions may influence the flow field around test models and because these conditions may often be approximated in the numerical simulations. This is particularly true for the transonic speed range where the flow is elliptic in nature, but it is also important for supersonic speeds where shock reflections or upstream influence through thick boundary layers may occur. Thus, carefully documented free-stream, wall-boundary, and downstream-boundary conditions are essential to the critical assessment of numerical simulations.

An example of the importance of including wall boundaries in the building-block approach for the study of a transonic flow is shown in Fig. 7. A low aspect ratio, nonlifting, swept wing was tested in a solid-wall tunnel. The wing was mounted in the center of the tunnel from the sidewall. Pertinent dimensions are shown in Fig. 7. Measured values of the pressure coefficients at several span locations are

compared with computations from a full-potential inviscid solution
(FLO29) using both free-air and solid-wall boundary conditions (Ref. 17).
(In the latter solution, account is taken of the upper-, lower-, and
far-wall boundaries.) Even though the wing-chord-to-tunnel-height ratio
is 3.75, the walls have a significant influence at this test Mach number,
and to properly assess the adequacy of this code, the appropriate bound-
ary conditions must be employed. Apparently, the comparison of data and
computation, including wall boundaries, indicates that the code is not
accounting for the tip effects present in the experiment. Further inves-
tigation regarding this comparison is under way.

Other examples and discussion of wall-boundary influence at tran-
sonic speeds may be found in Refs. 3 and 18.

Although it may seem obvious, it is worth noting that an accurate
description of model surface geometry is also very important. Analyti-
cally rather than numerically described shapes are always preferable and
are recommended whenever possible.

BUILDING-BLOCK EXPERIMENTS

Status and Plans

Experiments keyed to the development of computational aerodynamics
are being performed at the NASA-Ames Research Center and at several
universities under research grants from the Center. A pictorial repre-
sentation of experiments designed primarily to provide turbulence model-
ing information heretofore lacking for complex aerodynamic flows, is
given in Figs. 8a and 8b. Following the building-block concept discussed
previously in Fig. 6, the experiments address various important aerodynamic

13

flow phenomena, for example, attached, separating-and-reattaching, and trailing-edge flows. The year of the experiment and completion date are shown and available references noted in parentheses. Along with each experiment a computer code is being developed that incorporates precise experimental geometry and boundary conditions. Emphasis has been directed to developing Navier-Stokes codes that solve the Reynolds-averaged form of the conservation equations.

As the dates indicate, experiments on attached two-dimensional flows have been completed. Comparisons of these data and others available in the open literature with boundary-layer computations have demonstrated that two-equation turbulence models (e.g., Ref. 38) are adequate for most problems of aerodynamic interest.

Two-dimensional separating, reattaching, and trailing-edge flows have received considerable attention recently. A number of these experiments are complete and the others are under way. Emphasis is now being directed to developing improved turbulence models that will provide better numerical simulation for these flows (see, e.g., Ref. 39). The program for three-dimensional flows is just beginning. Many of the first experiments are attempts to set up appropriate test flows. This important flow regime needs considerably more attention in the near future, for computational capability is rapidly developing without the necessary experimental support.

Synergism of the Building-Block Concept

Examples of the synergistic process of combining experimental and computational studies have emerged from the building-block program

14

outlined above. Most notable of these is the realization that the Reynolds-averaged Navier-Stokes codes can already be used to predict unsteady phenomena, such as buffet caused by shock-induced separation and aileron buzz. It is of interest to note how this came about.

One of the first building-block experiments on shock-induced separation was a study of the transonic flow over a thick circular-arc airfoil (Ref. 23). Computational aerodynamics was used to assist in the design of the experiment, as shown in Fig. 9. Solutions for the free-air flow over airfoils of various thicknesses were computed using a Navier-Stokes code, but with the viscous terms inoperative. The primary motivation of the experiment was to study shock-induced separation, and the predictions of the local Mach number ahead of the shock wave for the 18%-thick section were in the regime expected to induce that type of separation. The upper and lower walls of the test section which were about one chord from the airfoil were constructed with the dimensions of the predicted free-air streamline, mainly to prevent the possibility of tunnel choking at the highest free-stream Mach numbers.

This successful experimental study provided several challenging flows for computations using the Navier-Stokes code, including the viscous terms (Ref. 23). Such codes are marched in time until a steady state is achieved. Figure 10 shows the Mach number-Reynolds number domain studied experimentally. The flows on either side of the shaded region were steady, having shock-induced separation that extended beyond the trailing edge at the highest Mach numbers and trailing-edge separation at the lower Mach numbers. Inside the region the flow was unsteady and periodic, alternating between shock-induced and trailing-edge

separation. Also, the lower Mach number boundary for the unsteady flow

changed when the Mach number was decreased continuously from the higher

values, indicating a hysteresis in the unsteady flow; that is, once it

was initiated it would persist to lower Mach numbers. Attempts to cor-

rectly calculate the high Mach number steady shock-induced separated

flow with a Navier-Stokes code that included the viscous terms were

unsuccessful (Ref. 23). The code was modified to include the wind-tunnel

wall shapes (Ref. 18), but the solution improvement was only minor. It

was then decided to attempt a computation at a lower Mach number

$(M_\infty = 0.754)$ where the tests for fixed tunnel Mach number resulted in

steady flow with a strong shock, but little separation (see Fig. 10);

this was done to see if the code could handle such cases. Convergence

of the solution, which was marched in time from a uniform state at free-

stream conditions, was determined by tracking the behavior of the pres-

sure coefficient with iteration number or in terms of chords traveled

by the flow. As shown by the solid line in Fig. 11, the solution did not

appear to be converging. After technical discussions — in which it

was pointed out that the experimental boundary of the unsteady

flow was very near the conditions of the computation, and that the

computation could actually be reproducing the unsteady flow — it was

decided to continue the calculations. As seen by the dashed curve in

Fig. 11, indeed, the computation did reproduce a periodic flow induced

by shock-boundary-layer separation (Ref. 18). A comparison of the

computed and measured pressures is shown in Fig. 12. This finding

resulted immediately in other related experimental and computational

studies. These have clearly demonstrated that buffet induced by shock-wave — boundary-layer interaction (Refs. 24, 25), and aileron buzz (Ref. 40) are predictable with these advanced Navier-Stokes codes.

An example of how the building-block concept has led to an improvement in turbulence modeling for shock-boundary-layer interactions is shown in Fig. 13. In the experiment, a shock wave was set up in an axisymmetric test section. Mean and fluctuating flow-field measurements have also been reported (Refs. 26, 27). Shown are the pressure and skin-friction data for a range of Reynolds numbers encompassing the practical range encountered by transonic airfoils. Calculations from a Reynolds-averaged Navier-Stokes code with several turbulence models are compared with the data. The pressure rise through the interaction is predicted, using any of the turbulence models. A more stringent test of the computations, however, is the comparison with the skin friction, c_f. The prediction using the two-equation turbulence model, which accounts better for the fact that the turbulence does not adjust immediately to changes in the mean flow (Ref. 38), is much better than the others. Flow-field profiles of mean velocity, turbulent shear, and turbulent kinetic energy are also predicted with reasonable accuracy with this model (Ref. 39).

VERIFICATION EXPERIMENTS

Only a limited number of verification experiments for complex flows are available; they are summarized, for the transonic speed range, in Ref. 3, in which they are evaluated and recommendations made for future experimental efforts. At this time, the NASA-Ames Research Center is

involved in cooperative efforts with McDonnell Douglas and Lockheed-Georgia for transonic tests and computations on supercritical wings. (This agreement is under the direction of L. L. Presley.) Some results from the McDonnell Douglas effort are given in Ref. 41. However, these tests need additional clarification regarding tunnel-wall boundary conditions; future experimentation on other shapes and different speed regimes is also needed.

Survey Recommendations

In attempting to plan a more extensive series of these verification experiments, an informal survey of industry and government research establishments was conducted several years ago. A questionnaire was used to obtain information on any new experiments that were planned and to obtain some consensus on what should be done in the future. The results of the survey are summarized in Table 3. The consensus generally followed that found subsequently by the AGARD Working Group 04 (Ref. 3). Some questionnaires were returned with additional narrative responses that indicated the following: Experimental redundancy is highly desirable; planning of experiments should involve experts in CFD and experimentation; and the experiments should be paced to the development of CFD, that is, they should progress from the simple to the complex.

CONCLUDING REMARKS

Computational Fluid Dynamics has been developing at a rapid pace. At this stage in its development, engineering applications are being made that employ the nonlinear inviscid flow equations with coupled

boundary-layer solutions. The next step in CFD development will be the use of the Reynolds-averaged Navier-Stokes equations, which have the potential of more general applications. However, this rapid development must be accompanied by a careful assessment through comparison with experiment.

A framework for complementary experimentation that can verify and critically assess the development of CFD has been outlined. Experiments have been categorized broadly as phenomenological and configurational, and it is proposed that they be keyed directly to the development stages of CFD codes. A building-block concept that couples experiment and computation was introduced. It is based on the idea of providing carefully documented modeling information for research-code development and carefully documented verification data for pilot codes that efficiently extend the research codes beyond their original demonstrative conditions. Building-block experiments, designed to supply turbulence modeling information for certain flow phenomena encountered in aerodynamic applications, were outlined. Some results from these experiments and their complementary computations were used to illustrate the synergism of this concept.
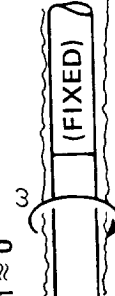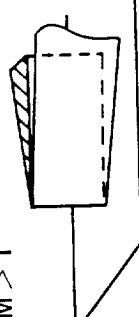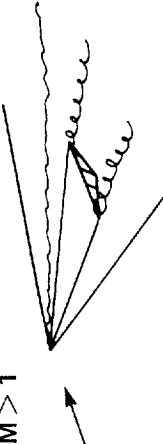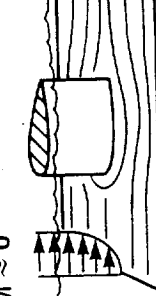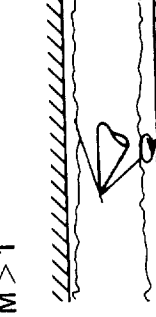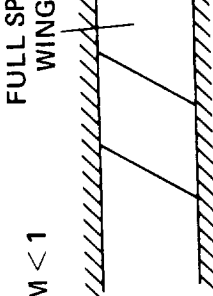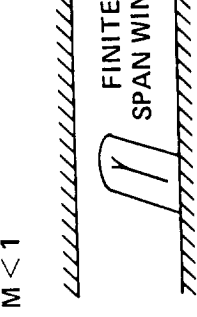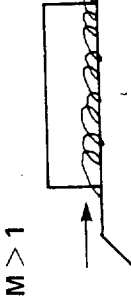
More work remains to be accomplished, especially in the area of three-dimensional flows. Very few building-block experiments are available, and emphasis on this particular phase of experimentation is strongly recommended.

REFERENCES

1. Chapman, D. R.: Dryden Lectureship in Research, Computational
   Aerodynamics Development and Outlook. AIAA Journal, vol. 17,
   no. 12, Dec. 1979, pp. 1293-1313.

2. Peterson, V. L.: Computational Aerodynamics and the Numerical
   Aerodynamic Simulation Facility. NASA CP-2032, 1977.

3. Experimental Data Base for Computer Program Assessment. AGARD
   Advisory Report No. 138, Report of the Fluid Dynamics Panel
   Working Group 04, May 1979.

4. Marvin, J. G.: Experiments Planned Specifically for Developing
   Turbulence Models in Computations of Flow Fields Around Aero-
   dynamics Shapes. AGARD CP-210, 1976, pp. 14-1 - 14-13.

5. Rakich, J. V.; and Lanfranco, M.: Numerical Computation of Space
   Shuttle Heating and Surface Streamlines. AIAA Paper 76-464,
   San Diego, Calif., 1976.

6. Cummings, J. W.; and Lockman, W. K.: Aerodynamic Heating Results
   for a Space Shuttle Orbiter 140B Model in the NASA/Ames 3.5-Foot
   Hypersonic Wind Tunnel (Test OH26). NASA CR-151,380, 1976.

7. Kacprzynski, J. J.; Ohman, L. H.; Garabedian, P. R.; and Korn, D. G.:
   Aeronautics Report LR-557, National Research Council of Canada,
   Ottawa, 1971.

8. Bauer, F.; Garabedian, P. R.; and Korn, D. G.: Lecture Notes in
   Economics and Mathematical Systems, vol. 108, Springer-Verlag,
   1972.

9. Bauer, F.; and Korn, D. G.: AIAA Paper 187-205, Hartford, Conn., 1975.

10. Melnik, R. E.: Wake Curvature and Trailing Edge Information Effects in Viscous Flow Over Airfoils. NASA CP-2045, 1978.

11. Wai, J.; and Yoshihara, H.: Viscous Transonic Flow Over Airfoils. Boeing Report D180-25934-1, 1980.

12. MacCormack, R. W.: An Efficient Explicit-Implicit-Characteristics Method for Solving the Compressible Navier-Stokes Equations. SIAM-AMS Proceedings of the Symposium on Computational Fluid Dynamics, New York, April 16-17, 1977.

13. Steger, J. L.: Implicit Finite Difference Simulation of Flow About Arbitrary Two Dimensional Geometries. AIAA Journal, vol. 16, no. 7, July 1978, pp. 679-686.

14. Deiwert, G. S.; and Bailey, H. E.: Prospects for Computing Airfoil Aerodynamics With Reynolds Averaged Navier-Stokes Codes. NASA CP-2045, 1978.

15. Ferziger, J. H.: Large Eddy Numerical Simulations of Turbulent Flows. AIAA Journal, vol. 15, no. 9, Sept. 1977, pp. 1261-1267.

16. Chapman, D. R.: Trends and Pacing Items In Computational Aerodynamics. 7th International Conference on Numerical Methods in Fluid Mechanics, Stanford University, Stanford, Calif./NASA-Ames Research Center, Moffett Field, Calif., June 23-27, 1980.

17. Lockman, W. L.: Private communication on NASA-Ames experiment and computations, 1980.

① ATTACHED FLOWS

M ≈ 0   (FIXED)   1978-81

M ≈ 0   1978-81

M < 1   FULL SPAN WING   1979 (35)

M > 1   1979-82

② SEPARATING/REATTACHING FLOWS

M > 1   1976 (36)

M > 1   1980 (37)

M < 1   FINITE SPAN WING   1979-80

M < 1   FULL SPAN WING   A/R ≈ ∞   1980-81

③ TRAILING/LEESIDE FLOWS

M > 1   1980-81

MACH AND REYNOLDS NO. DOMAINS FOR AERODYNAMIC FLOWS

COMPUTATION

WALL SHAPE FROM
COMPUTATION

NOMINAL
TEST
CONDITION

SHOCK-
INDUCED
SEPARATION

t/c

0.18
0.15
0.12

$\frac{y_s}{h} < \frac{2}{3}$

M_1

1.6

1.4

1.2

1.0

.6   .7   .8   .9

M_∞

h

y_s

M_1

c = 20.3 cm
(8 in.)
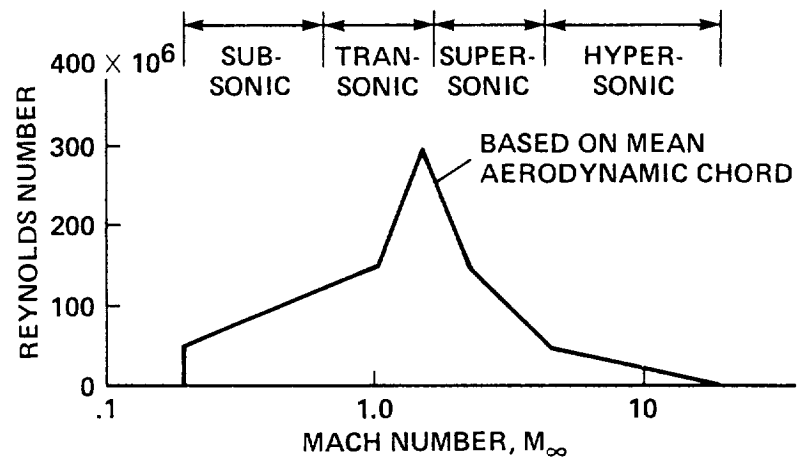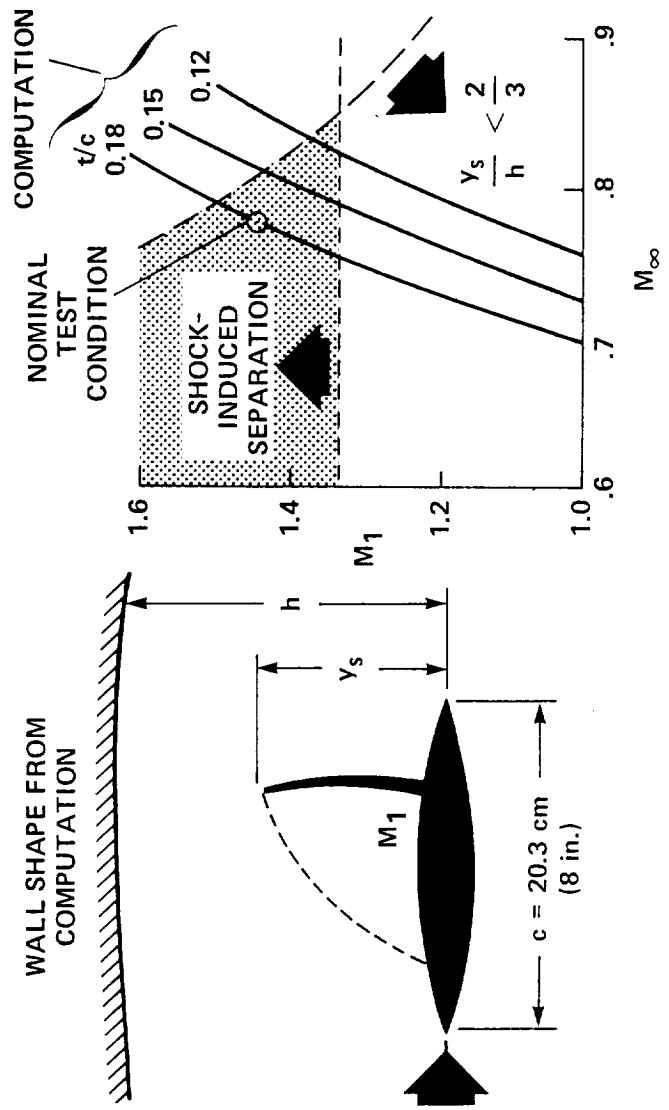
ACCELERATING FLOW
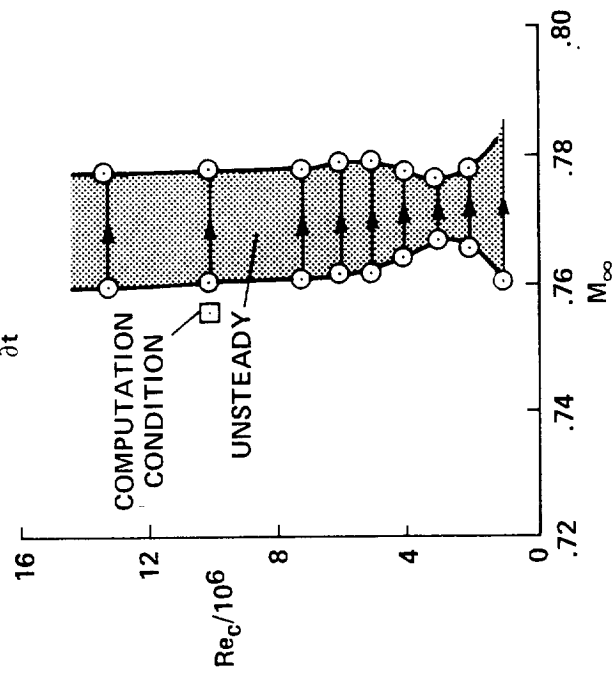
$$\frac{\partial M_\infty}{\partial t} = 0.001$$

DECELERATING FLOW

$$\frac{\partial M_\infty}{\partial t} = -0.001$$

COMPUTATION q_o = 25.1 watts/cm^2
EXPERIMENT T_W/T_t = 0.26
               Re_L = 1.7 x 10^6
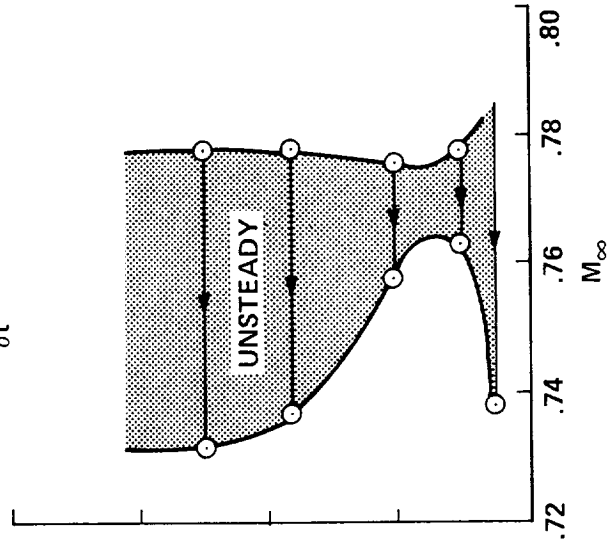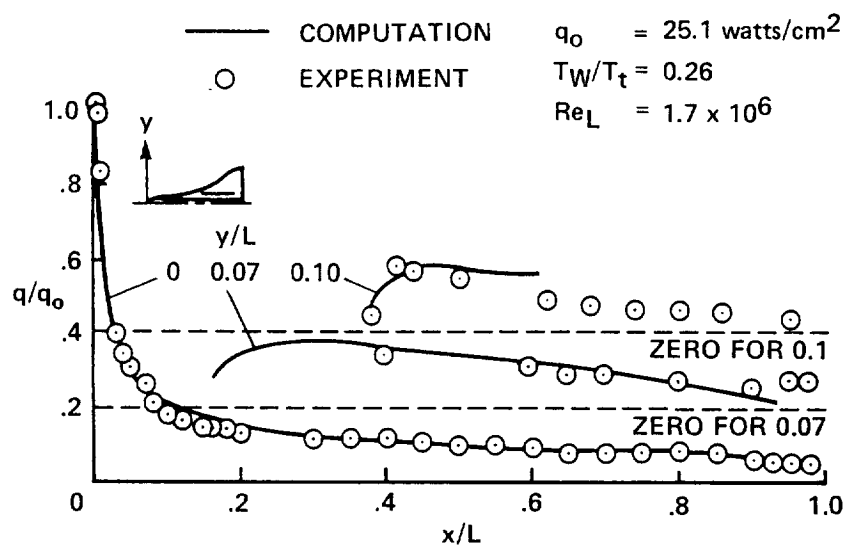
COMPUTATION

FLIGHT $\alpha = 32°$, $u_\infty = 5.1$ km/sec, $q_0 = 35$ watts/cm^2

WIND TUNNEL $\alpha = 30°$, $M_\infty = 7.3$, $q_0 = 25$ watts/cm^2

FLIGHT EXTRAPOLATION

WIND TUNNEL
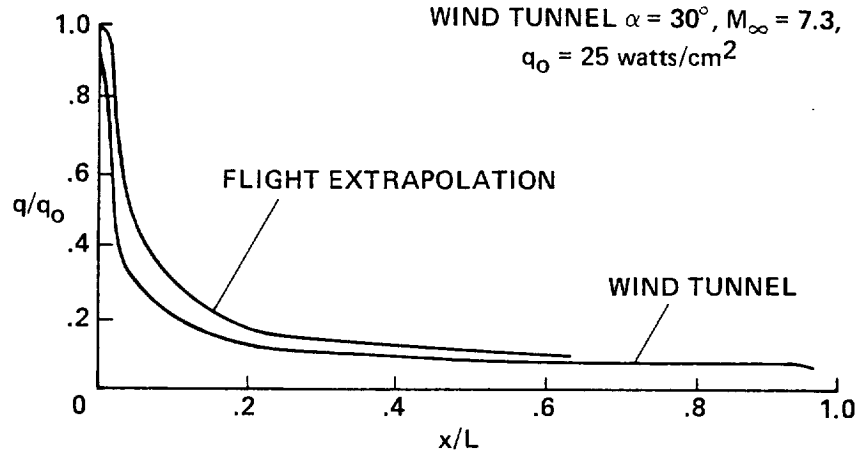
	C_D	C_L	
○ EXP	0.037	0.87	REF. 7
- - - VISCOUS	0.024	0.97	REF. 9
—— INVISCID	0.064	1.24	REF. 8
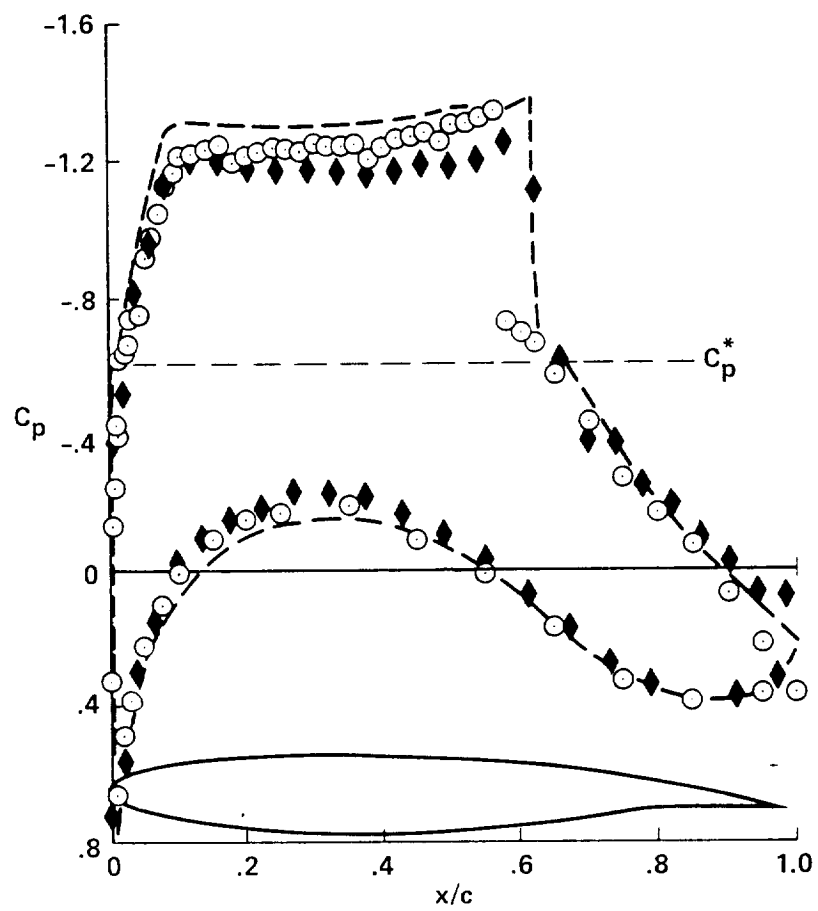
DRAG POLAR
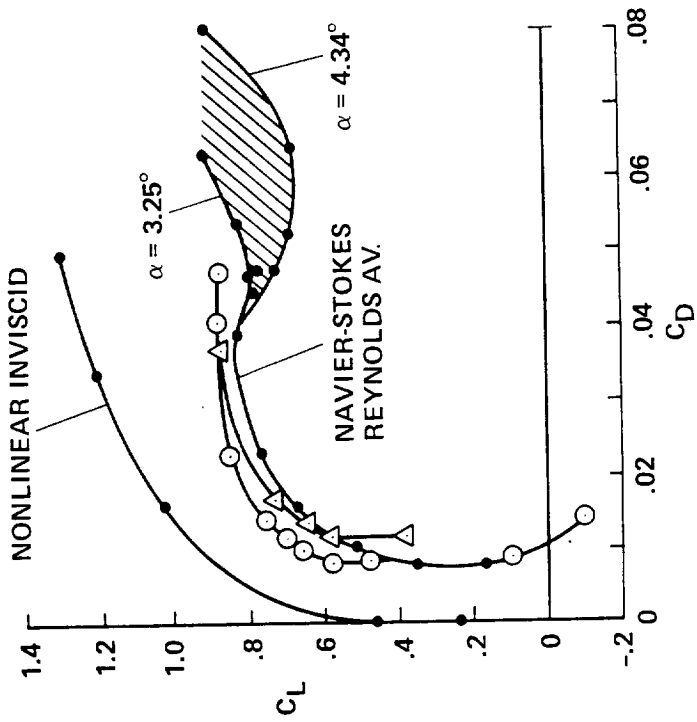
LIFT CURVE

EXPERIMENT

○ 22.5% TUNNEL WALL POROSITY
△ 6% TUNNEL WALL POROSITY

NAVIER-STOKES
REYNOLDS AV.

BUFFET DOMAIN
(COMPUTED)

ANGLE OF ATTACK, deg

c_L

NONLINEAR INVISCID

$\alpha = 3.25°$

$\alpha = 4.34°$

NAVIER-STOKES
REYNOLDS AV.

c_D

PRODUCTION
CODES

PILOT
CODES

DESIGN
EXPERIMENTS

(configurational)

RESEARCH
CODES

VERIFICATION
EXPERIMENTS

(parametrical)

BUILDING-BLOCK
EXPERIMENTS

(phenomenological)
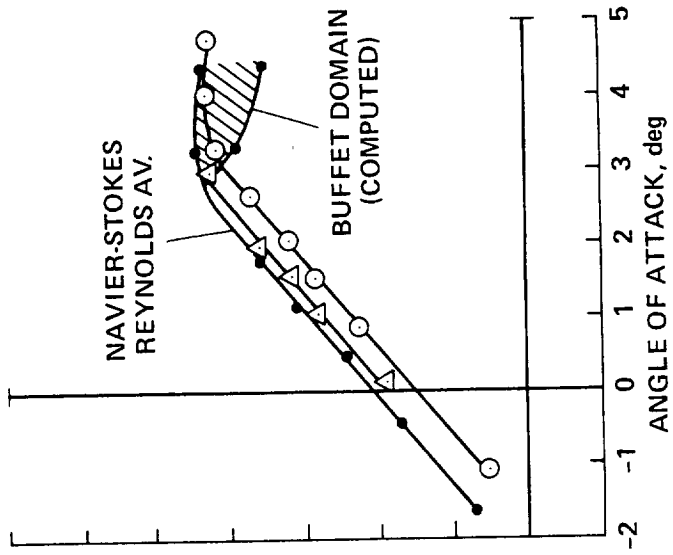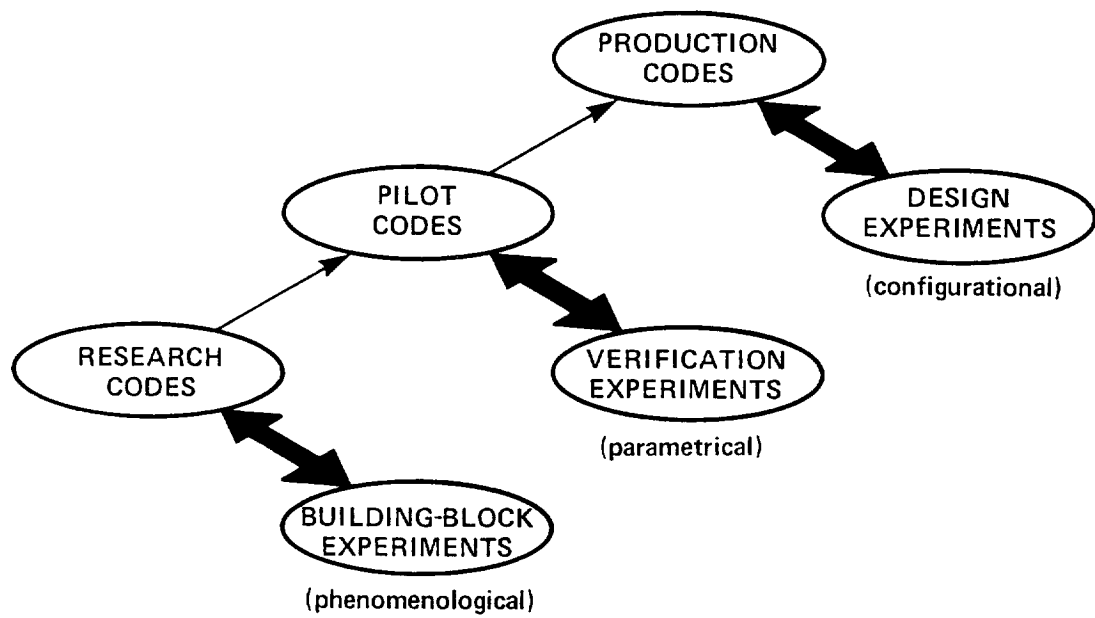
NACA 0012 STREAMWISE SECTION

$\Lambda = 20°$ $AR = b/c = 3$ $\lambda = 1$

$H/c = 3.75$ $W/\lambda = 2.5$ $(b/2)/W = 0.6$

O   EXPERIMENT, REF. 17

$Re_c = 8.07 \times 10^6$

——— WITH WALL BOUNDARIES

- - - - FREE AIR

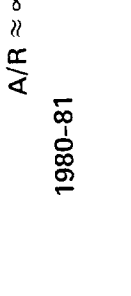FLO-29 INVISCID CODE

$M_\infty = 1.4$ $\delta_o \approx 2.5$ cm
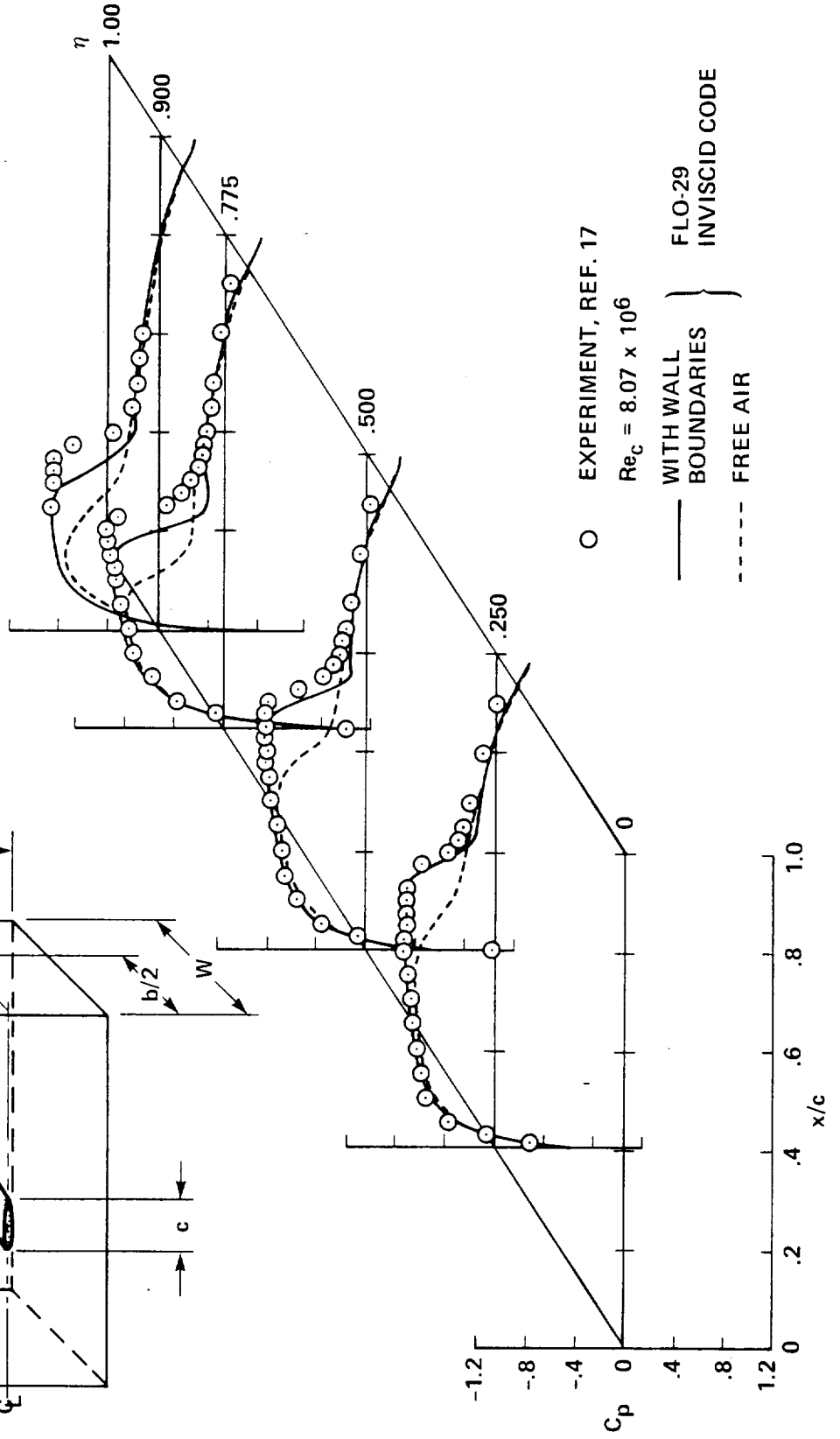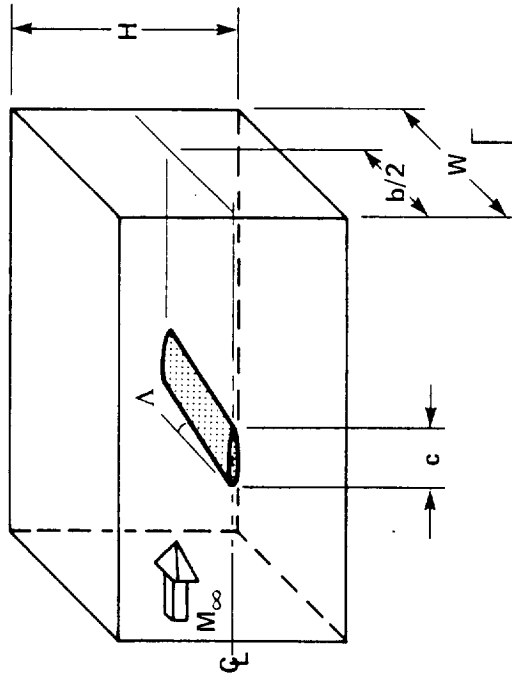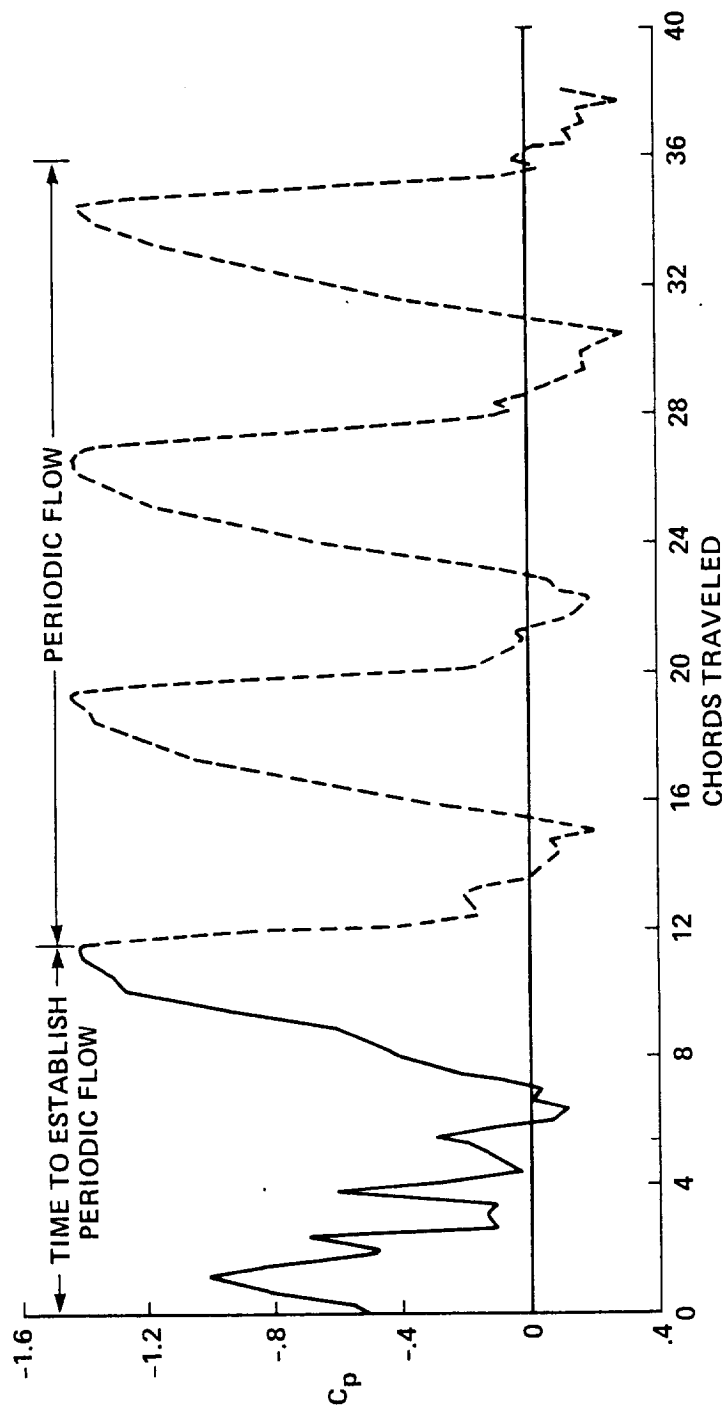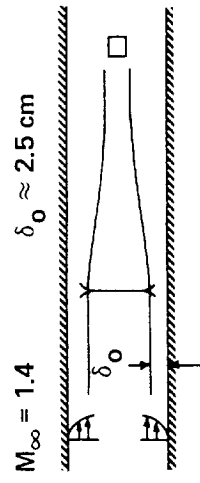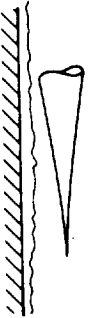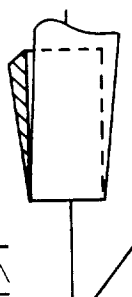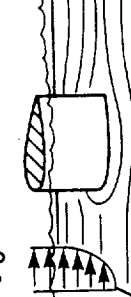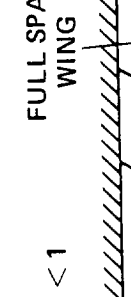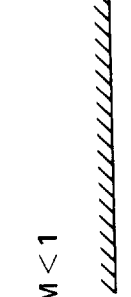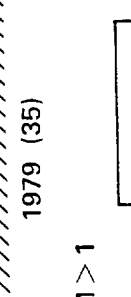
○ EXPERIMENT
— 0 – EQ. MODEL (1970)
– – 1 – EQ. MODEL (1976)
– · – 2 – EQ. W-R MODEL (1978)

$Re_{\delta_o} = 0.12 \times 10^6$

$Re_{\delta_o} = 0.5 \times 10^6$

$Re_{\delta_o} = 5.4 \times 10^6$

PRESSURE, p_w/p_∞

SKIN FRICTION, c_f

$(x-x_o)/\delta_o$

① ATTACHED FLOWS

M > 1
1969 (REF. 19)

M >> 1
1972 (20)

M > 1
1973 (21)

M > 1
1976-77 (22)

② SEPARATING/REATTACHING FLOWS

M < 1
1975-78 (23, 24, 25)

M > 1   M < 1
1975-78 (26, 27)

M > 1
1975-76 (28)

M < 1
1979-80 (29, 30)

M > 1
1978 (31)

M < 1
1980-81

M ≈ 0
1980 (32)

$M_\infty = M_{\sin \omega t}$
M ≈ 0
1979-81

③ TRAILING-EDGE FLOWS

M ≈ 0   ROUGHNESS
1978-81

M ≈ 0
1978-81

M < 1
1978-81 (33, 34)

① ② ③
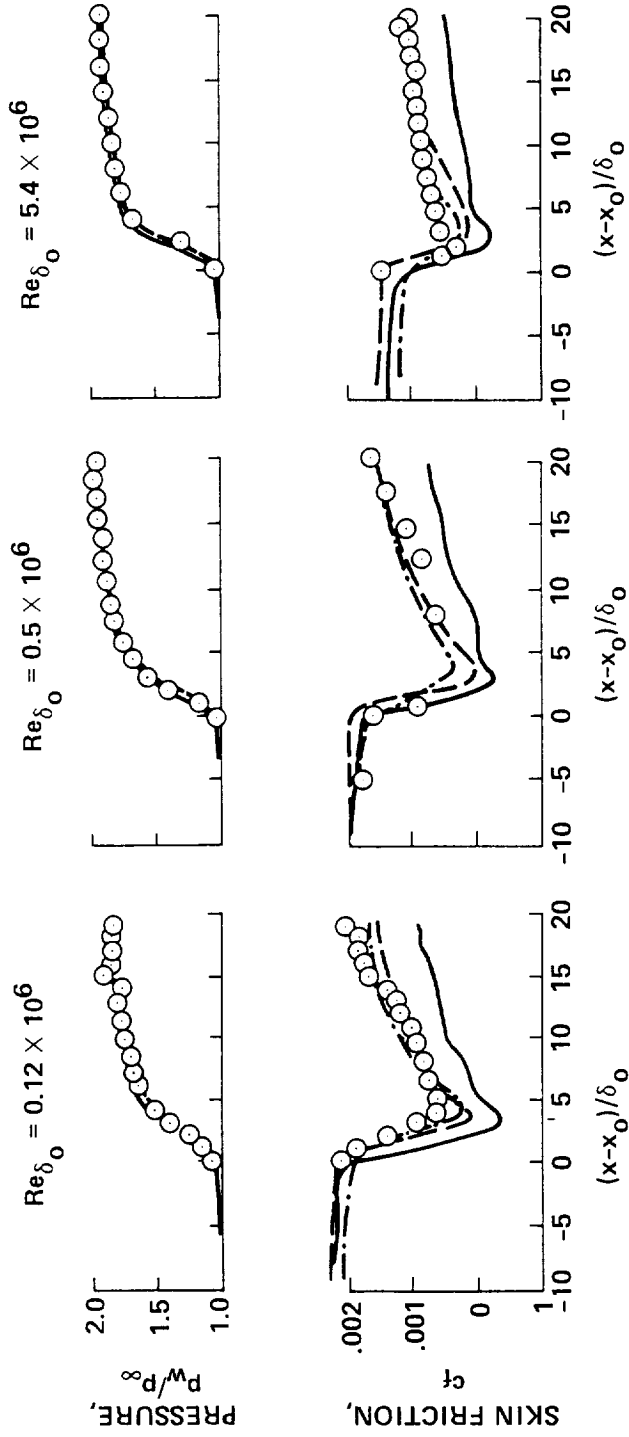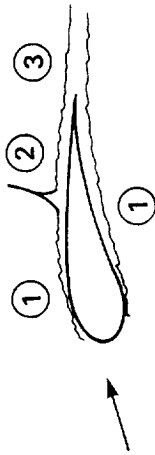
① ATTACHED FLOWS

② SEPARATING/REATTACHING FLOWS

③ TRAILING/LEESIDE FLOWS

M ≈ 0

(FIXED)

1978-81

M ≈ 0

1978-81

M < 1

FULL SPAN WING

1979 (35)

M > 1

1979-82

M > 1

1976 (36)

M > 1

1980 (37)

M < 1

FINITE SPAN WING

1979-80

M < 1

FULL SPAN WING A/R ≈ ∞

1980-81

M > 1

1980-81